

SIMULATION OF NATURAL CONVECTION HEAT TRANSFER IN AN
ENCLOSED CAVITY USING CONSTRAINED INTERPOLATED PROFILE
NAVIER-STOKES EQUATION (CIPNSE) METHOD

LOH CHEE KIN

A thesis submitted in fulfillment of the
requirements for the award of the degree of
Master of Engineering (Mechanical)

Faculty of Mechanical Engineering
Universiti Teknologi Malaysia

NOVEMBER 2010

To my beloved family

ACKNOWLEDGEMENTS

First of all, I would like to express my sincerest appreciation to my supervisor Dr. Nor Azwadi Che Sidik for all his guidance and supports throughout the whole research and thesis writing. His advices and suggestions really helped in this project and I had learned a lot from him. Through this Master project, I have gained much knowledge as well as many experiences that are useful for my future.

Besides, I would like to thank my family especially my parents for their support by giving me advice to continue this research.

Finally, I would like to thank my fellow friends who always encourage me and being helpful. Thank you for all your moral support.

ABSTRACT

Constrained Interpolated Profile Navier-Stokes Equation (CIPNSE) method was introduced to simulate characteristic of the natural convection heat transfer mechanism and the nature of fluid flow exist in an enclosed square cavity with differentially heated side walls. The fundamental idea of this study is to solve the advection phase equation with CIP method and the non-advection phase equation is calculated with finite difference method. CIPNSE scheme solves hyperbolic equations for vorticity transport equation and the energy equation. The application of CIPNSE is to predict the phenomena of fluid flow in the cavity due to the effect of various Rayleigh number, $Ra=10^3$, 10^4 and 10^5 . The results obtained were in streamline and isotherms at different Rayleigh number of the flow in the cavity. Corresponding results show a good agreement with the established results by C.S.N Azwadi and M.S.Idris. This numerical approach is found to be good in stability for solving the natural convection heat transfer problem effectively.

ABSTRAK

Kaedah '*Constrained Interpolated Profile Navier-Stokes Equation (CIPNSE)*' diperkenalkan untuk mensimulasikan ciri-ciri mekanisme pemindahan haba konveksi secara semulajadi dan sifat aliran bendalir yang ada dalam rongga persegi tertutup dengan suhu dinding sisinya yang berbeza. Idea asas dalam penelitian ini adalah untuk menyelesaikan persamaan fasa adveksi dengan kaedah CIP dan persamaan fasa tak adveksi dikira dengan kaedah pembeza terhingga. Skim CIPNSE menyelesaikan persamaan hiperbolik bagi persamaan pengangkutan pusaran dan persamaan tenaga. Penerapan CIPNSE adalah untuk menjangkakan fenomena aliran cecair dalam rongga akibat kesan pengaruh pelbagai nombor Rayleigh, $Ra = 10^3$, 10^4 dan 10^5 . Keputusan yang diperolehi dalam bentuk *streamline* dan *isotherm* dengan nombor Rayleigh yang berbeza dalam pengaliran rongga. Keputusan tersebut menunjukkan kesepakatan yang baik dengan keputusan yang diperolehi C.S.N. Azwadi dan M.S.Idris. Pendekatan berangka didapati mantap dalam kestabilan bagi menyelesaikan masalah perpindahan haba konveksi secara semulajadi dengan berkesan.

TABLE OF CONTENTS

CHAPTER	TITLE	PAGE
	DECLARATION	ii
	DEDICATION	iii
	ACKNOWLEDGEMENTS	iv
	ABSTRACT	v
	ABSTRAK	vi
	TABLE OF CONTENTS	vii
	LIST OF FIGURES	x
	LIST OF ABBREVIATIONS	xi
	LIST OF SYMBOLS	xii
	LIST OF APPENDICES	xiv
1	INTRODUCTION	
	1.1 Background of Study	1
	1.2 Problem of Statement	4
	1.3 Objective	4
	1.4 Scopes of Study	5
2	GOVERNING EQUATIONS	
	2.1 Constrained Interpolated Profile (CIP)	6
	2.2 The Governing Equations	7
	2.3 Accuracy of Finite Difference Solutions	10
	2.4 Grashof Number, Gr	11
	2.5 Prandtl Number, Pr	11

2.6	Rayleigh Number, Ra	12
2.7	Governing Equations in Square Cavity Flow	13
2.8	Dimensionless Variables	15
2.9	Derivation of Dimensionless Equations	16
2.9.1	Vorticity Transport Equation	16
2.9.2	Vorticity Equation in Term of Stream Function	19
2.9.3	Energy Equation	21
2.10	Discretization	23
2.11	Boundary Conditions	25
2.12	Grid Generation	29
2.13	One Dimensional CIP Hyperbolic Equation	30
2.14	Constrained Interpolated Profile Navier Stokes Equation (CIPNSE)	31
2.14.1	CIPNSE for Vorticity Transport Equation	32
2.14.2	Non-advection Phase of Vorticity Transport Equation	33
2.14.3	CIPNSE for Energy Equation	37
2.14.4	Non-advection Phase of Energy Equation	38
2.14.5	The Advection Phase	43
3	METHODOLOGY	
3.1	Algorithm for the Numerical Simulation	49
3.2	Convergence Criteria for CIPNSE Method	49
3.3	Flow Chart	50

4	RESULT AND DISCUSSION	
4.1	Results	51
4.2	Streamline Plots of Various Rayleigh Number at Ra=10 ³ , Ra=10 ⁴ and Ra=10 ⁵	52
4.3	Isotherms Plots of Various Rayleigh Number at Ra=10 ³ , Ra=10 ⁴ and Ra=10 ⁵	55
4.4	Comparison Result of CIPNSE Method with Lattice Boltzmann Method (LBM)	57
5	CONCLUSION AND RECOMMENDATION	
5.1	Conclusion	61
5.2	Recommendation	62
	REFERENCES	63
	Appendices A-B	66-77

LIST OF FIGURES

FIGURE NO	TITLE	PAGE
2.1	Boundary condition of a square cavity flow	25
2.2	Square meshing of the cavity	29
2.3	Corresponding of CIP method for first order wave equation with several classical methods	31
2.4	Meshing in two dimensional CIP	45
3.1	Schematic geometry diagram of natural convection in a square cavity	48
3.2	Flowchart for CIPNSE method	50
4.1	Streamline at $Ra=10^3$	52
4.2	Streamline at $Ra=10^4$	53
4.3	Streamline at $Ra=10^5$	54
4.4	Isotherms at $Ra=10^3$	55
4.5	Isotherms at $Ra=10^4$	55
4.6	Isotherms at $Ra=10^5$	56
4.7	Comparison of streamline plots for various Rayleigh number between CIPNSE and LBM	57
4.8	Comparison of isotherms plots for various Rayleigh number between CIPNSE and LBM	59

LIST OF ABBREVIATIONS

CIP	-	Constrained Interpolated Profile
CFD	-	Computational Fluid Dynamics
PDE	-	Partial Differential Equation
NSE	-	Navier-Stokes Equation
LBM	-	Lattice Boltzmann Method
FDM	-	Finite Difference Method
FVM	-	Finite Volume Method
FEM	-	Finite Element Method
CIPNSE	-	Constrained Interpolated Profile Navier-Stokes Equation Method

LIST OF SYMBOLS

p	-	Pressure
ρ	-	Density
g	-	Gravitational acceleration
L	-	Length
β	-	Volumetric thermal expansion
T_h	-	Surface wall with hot temperature
T_c	-	Surface wall with cold temperature
C_p	-	Specific heat
k	-	Thermal conductivity
H	-	Length of cavity
t	-	Time
τ	-	Dimensionless time
u	-	Velocity in x direction
U	-	Dimensionless velocity in x direction
v	-	Velocity in y direction
V	-	Dimensionless velocity in y direction
x	-	Axial distance
X	-	Dimensionless axial distance
y	-	Vertical distance
Y	-	Dimensionless vertical distance
T	-	Temperature

Dimensionless Parameters

- AR - Aspect Ratio
- Gr - Grashof Number
- Pr - Prandtl Number
- Ra - Rayleigh Number

Greek Symbols

- θ - Dimensionless temperature
- μ - Dynamic viscosity
- ν - Kinematic shear viscosity
- α - Thermal diffusivity
- ω - Vorticity
- Ω - Dimensionless vorticity
- ψ - Stream function
- Ψ - Dimensionless stream function
- ∇ - Nabla operator

Superscript

- n - Current value
- $n + 1$ - Next step value
- $*$ - Non advection phase value

Subscript

- i - x direction node
- j - y direction node
- $max\ i$ - x direction maximum node
- $max\ j$ - y direction maximum node

LIST OF APPENDICES

APPENDIX	TITLE	PAGE
A	Codes for CIPNSE of streamline plots square cavity flow	66
B	Codes for CIPNSE of isotherms plots square cavity flow	72

CHAPTER 1

INTRODUCTION

This chapter begins with the introduction about the background of the study, problem of statement, objective and finally the scope of study of this project. The following chapters are consisted by governing equations, methodology, result and discussion, conclusion and recommendation.

1.1 Background of Study

Recently, Computational Fluid Dynamic (CFD) has been widely used in many approaches due to the existence of the computer technology bringing the greatest benefits to the researcher. Over the years, a lot of studies have been carried out for generating a better approximating result in terms of effectiveness and efficiency. Although the researchers would like to prefer CFD as the alternative tool to describe the fluid flow behavior, but the error gain is found that will be an issue in the simulations. This problem needs to be concerned for better solutions in case the improvement of computational method is able to produce high resolution. Basically, CFD is classified as the numerical methods and algorithm in solving problem and also obtaining the solutions which is related to fluid flows of fluid mechanics. The physical principle can be expressed in terms of mathematical equations that the partial differential equation (PDE) is widely described the Navier-stokes equations as to be the fundamental of CFD fluid flow simulation [1]. Based on the conventional CFD, it is not sufficient in solving

the multiphase flows with a complicated physical phenomenon due to the complexity of the partial differential equation [1]. Therefore, there are several numerical methods including finite difference method (FDM), finite volume method (FVM), finite element method (FEM) and so on can be applied in solving the fluid flow problem [2].

In general, there are three alternative approaches or methods that can be applied in solving the fluid mechanics and heat transfer problem. These approaches can be used are:

1. Computational Fluid Dynamics (CFD)
2. Analytical
3. Experimental

Computational Fluid Dynamics or CFD normally uses advanced computer software to model the flow of fluids through a processing facility. There are few states that the fluid may be in liquid, gas or loose particle form, or a combination of them. Applying computer simulation a wide range of variations in physical design and operational parameters can be tested and refined until a set which gives optimum performance is identified. Computational fluid dynamics contribute as the “third approach” in the study and development of the fluid dynamics. There are many numerical simulations of applying CFD have been giving greatest implication for researchers and scientist. CFD can be classified as a research tool which is able to solve the problem numerically. Furthermore, computer program has to be developed for solving the flow problems in the numerical simulation. In fact, CFD codes are structured around the numerical algorithms that will be an alternative way in solving the fluid flow problems [3].

Computational Fluid Dynamics, CFD provides a qualitative prediction fluid flow by several techniques, there are mathematical modeling, numerical methods and software tools. CFD leads the researchers and engineers to perform the numerical experiments also called as computer simulations in a particular virtual flow laboratory.

Computational fluid dynamics has become increasingly important in hydrodynamic stability as the numerical analysis has improved with new technology as the computers have become faster and gained more memory, so that the Navier-Stokes equations may be integrated accurately for many kinds of fluid flows. Indeed, computational fluid dynamics has now reached a position where it can compare with laboratory investigation of hydrodynamic stability by typical simulation.

Nowadays, computational fluid dynamics is having an equal partner with pure theory and pure experiment in the analysis and also solution of fluid dynamic problems, As long as our advanced human civilization exists, CFD will play an important role in solving the fluid problem.

The theoretical method is often referred to an analytical approach meanwhile the terms computational and numerical are used interchangeably. Analytical approach is one of the three methods can be apply in obtaining the solution of fluid flow problem. Analytical models are categorized as mathematical models that have a closed form solution whereby the solution to the equations used to describe the changes in a system can be expressed as mathematical function.

Firstly, an equipment model would need to be built and constructed in the experimental approach. Although the experimental approach has its own capability in providing more realistic solutions for many flow problem, but the cost consuming arises while in getting source and the equipments.

Natural convection heat transfer in the enclosed cavity has received much attention in recent years. There are many types of flows having a wide range of applications in aerodynamics, pipe equipment components and so on. In natural convective heat transfer, heat is transferred between a solid surface and a fluid moving across it. The fluid motion has a relationship with the buoyancy forces which will determine the nature of the fluid motion. Buoyancy force arises due to the changes of density that result from the temperature variations in the fluid. It is found that the natural

convection of the fluid flow or movement can be either laminar or turbulent flow. However, the chances of laminar flow occurs is higher than the turbulent flow is because of the low velocities usually exist in natural convection. Thus, it is necessary to gain more understanding on the natural convection heat transfer in the enclosed cavity. There are many researches based on the elementary idea and interest for studying the heat transfer mechanism [4-6] and also fluid flow behavior on the cavity wall [7-8].

1.2 Problem of Statement

Recently the issue occurs due to the conventional numerical methods in obtaining solution of Navier-Stokes equation are not sufficient for the complicated system. That is due to poor stability condition for numerical method when dealing with the higher order of accuracy of the mesh size grid will definitely give sense to the result.

1.3 Objective

The objective of this project is to extend the formulation of Constrained Interpolated Profile (CIP) method for Navier-Stokes to predict thermal fluid flow behavior and to study the thermal fluid flow behavior in a differentially enclosure walls for better understanding.

1.4 Scopes of Study

- 1 Solving the advection equation with the application of CIP method
- 2 Comparison of the computed results with those available in literature
- 3 The test case is the thermal fluid flow in a differentially heated square enclosure wall
- 4 The flow is considered as incompressible and laminar